A PRESSURE-BASED SOLVER FOR COMPRESSIBLE THREE-PHASE FLOW WITH PHASE CHANGE

BINGSHENG YE[1,2], YIWEI WANG*[1,2], CHENGUANG HUANG[1],JIAN HUANG[1]
[1] *Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, China*
{yebingsheng, wangyw , huangcg , huangjian }@imech.ac.cn
[2] *School of Engineering Science, University of Chinese Academy of Sciences, Beijing, 100049, China*

Keywords: *Cavitating Flow, Shedding Mechanism, Shock Propagation*

Introduction

Cavitation is a common phenomenon capturing the attention of industry and academia. In general, it results in negative and undesirable effect on structure, like erosion on structure. To reduce or avoid damage caused by cavitation, research works have been carried out for a long period of time, establishing several theories. Since a few practical solvers which consider the effect of phase change are available in OpenFOAM, most problems involving cavitation can be investigated through numerical method. However, recent research progress and difficulty in engineering introduce new requirements. On the one hand, cavitation near free surface has become a heat topic in the field of high speed hydrodynamics, requiring a solver able to deal with three phases, including liquid(water) and other two kinds of gas (vapor and non-condensable gas), as well as considering effect of phase change. On the other hand, in the interesting work of Genesh et al [1] a high temporal resolution X-ray device was used to investigate the shedding mechanism of cavity by measuring density inside, which proposes the concept of cavity shedding induced by shock, quite different from previous agreement that re-entry jet dominates the process of cavity shedding. To reflect the propagation of shock in numerical simulation, compressibility should be taken into consideration. Since none solver available can satisfy the demands above, a new useable one is in urgent need.

Method

A pressure-based solver for compressible three-phase flow with phase change is developed based on the utility of *interPhaseChangeFoam* and *compressibleInterFoam*. Besides the effect of free surface and shock propagation in cavitating flow mentioned above, many other relevant research can be also conducted with this new solver.
Solvers can be density-based or pressure-based. The density-based solvers were widely used in hypersonic problems in aviation. But they would have difficulty when Mach number is not that high. Within the frame of OpenFOAM, *cavitatingFoam* is a density-based solver which consider phase change, but quite unstable in subsonic condition. In several cases tested iteration cannot obtain convergence unless a tiny time step is adopted which however leads to another trouble that calculation will never be finished. This situation is due to stiffness of matrix given by density-based algorithm when Mach number is low. Although it can be overcome with some specialized solvers implementing pre-conditioning approach [2], we still prefer to pressure-based solvers which are much more stable itself in many tests. Pressure-base method is firstly designed to deal with incompressible flows. But recent research has expanded their application to flows at all speeds [3]. Fortunately, most solvers considering compressibility in OpenFOAM are pressure-base, providing convenience to establishing a new solver for our requirement.
A few solvers in OpenFOAM are able to deal with a system of more than two phases. A member function named *MULES:correct* is first called to calculate the flux of volume fraction of each phase, followed by use of *MULES:limitSum* to adjust that flux so that in the next step where the volume fraction transport equation of each phase is solved by *MULES:explicitSolve* the sum can be ensured to be nearly 1. Some tests indicate that this method will encounter some difficulties if phase change is involved, e.g. much less cavity than experimental observation. So in our solver only two phases, water and non-condensable gas, are consider to have independent volume fraction, namely α_{vapor} is obtained through the constraint $\sum \alpha_i = 1$.

Results and discussion

A case about cavitating flow around an axisymmetric projectile is conducted by this new solver, also compared with the numerical result from *interPhaseChangeFoam* as well as the experimental result based on the SHPB (Split Hopkinson Pressure Bar) technology and high-speed photography. The inflow velocity is 18.5m/s with a cavitation number about 0.572. A 2D axisymmetric mesh is used. Figure 1 shows the comparison of variety of cavity shape over time， reflecting good agreement with each other, especially in the first half cycle. The length of cavity versus time is also measured, indicating that whether the compressibility is considered or not, the numerical results almost coincide but a shift exists when they are compared with the experimental result.

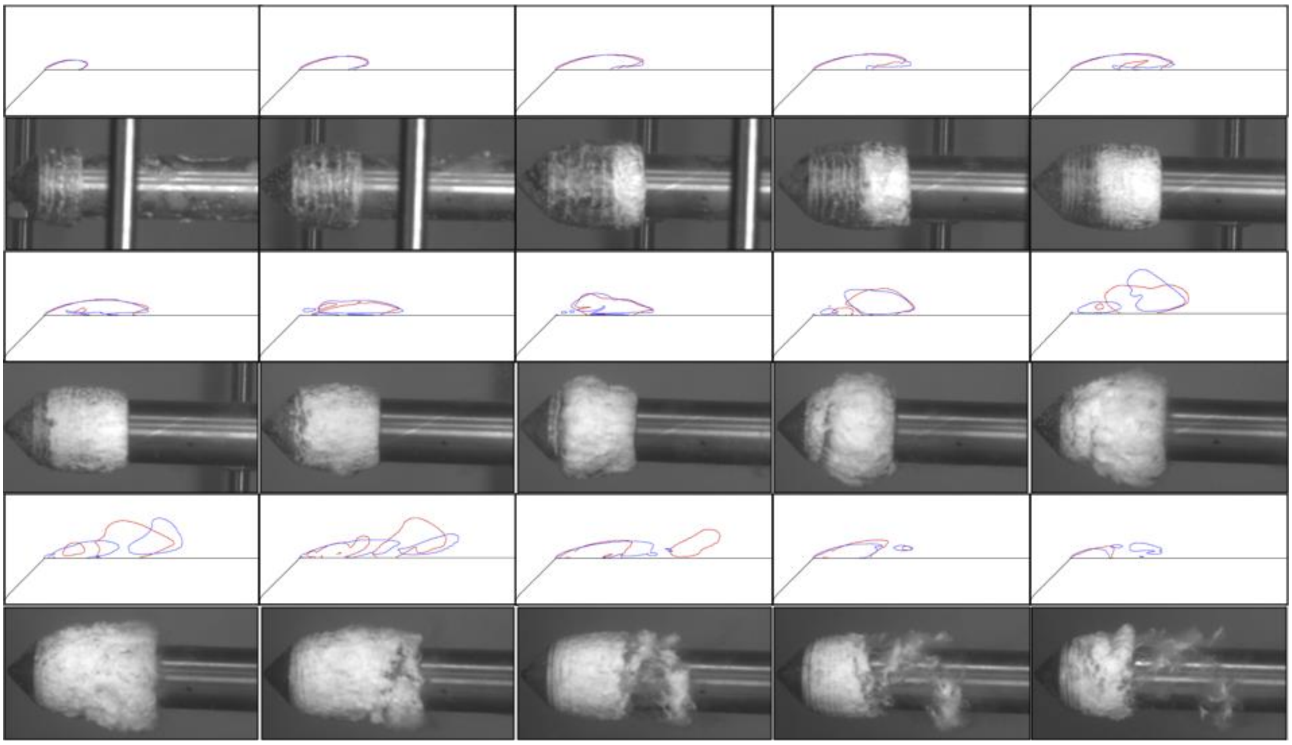

Figure 1: Time sequences of cavity shape obtained from a new solver (red lines), *interPhaseChangeFoam* (blue), compared with experimental observation

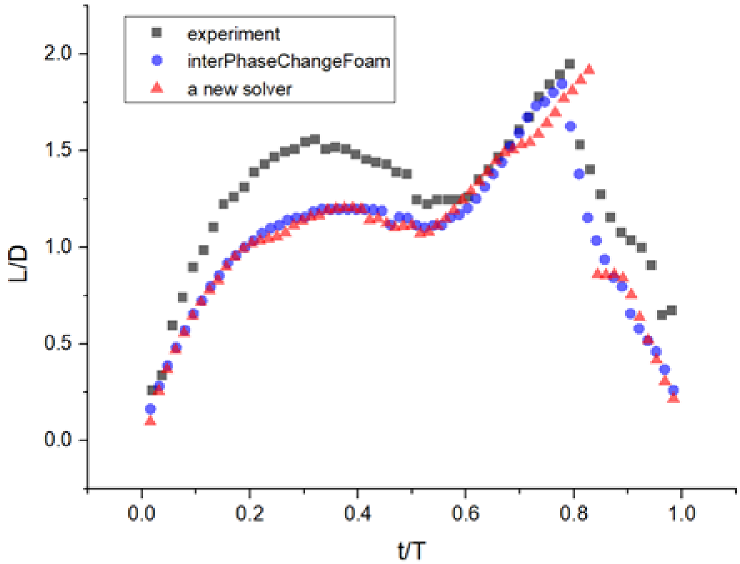

Figure 2: Comparison of cavity length

Most previous research works were focused on the shedding mechanism induced by the re-entry jet, which is usually considered as the most important factor on the transition [4]. Recently proposed concept of shedding induced by shock propagation has impelled academia to reassess the mechanism in cavitation instability. In our results, different mechanisms above is found. In the first half cycle, the re-entry jet whose formation is promoted by the inverse pressure gradient afterwards is attached to trailing edge of the cavity and finally cut it off. However, the last half cycle seems to be quite different. A part of the cavity shedding previously collapses, leading to formation and propagation of a shock. When intersected by high pressure of the shock, the rest part also begins to collapse from its closure as shown by high condensation rate in Figure 3. The velocity field indicates that although the re-entry jet still generates, its head has a distance from trailing edge of the cavity therefore contribute little in shedding process. So it is more believable that the last half is dominated by effect of shock.

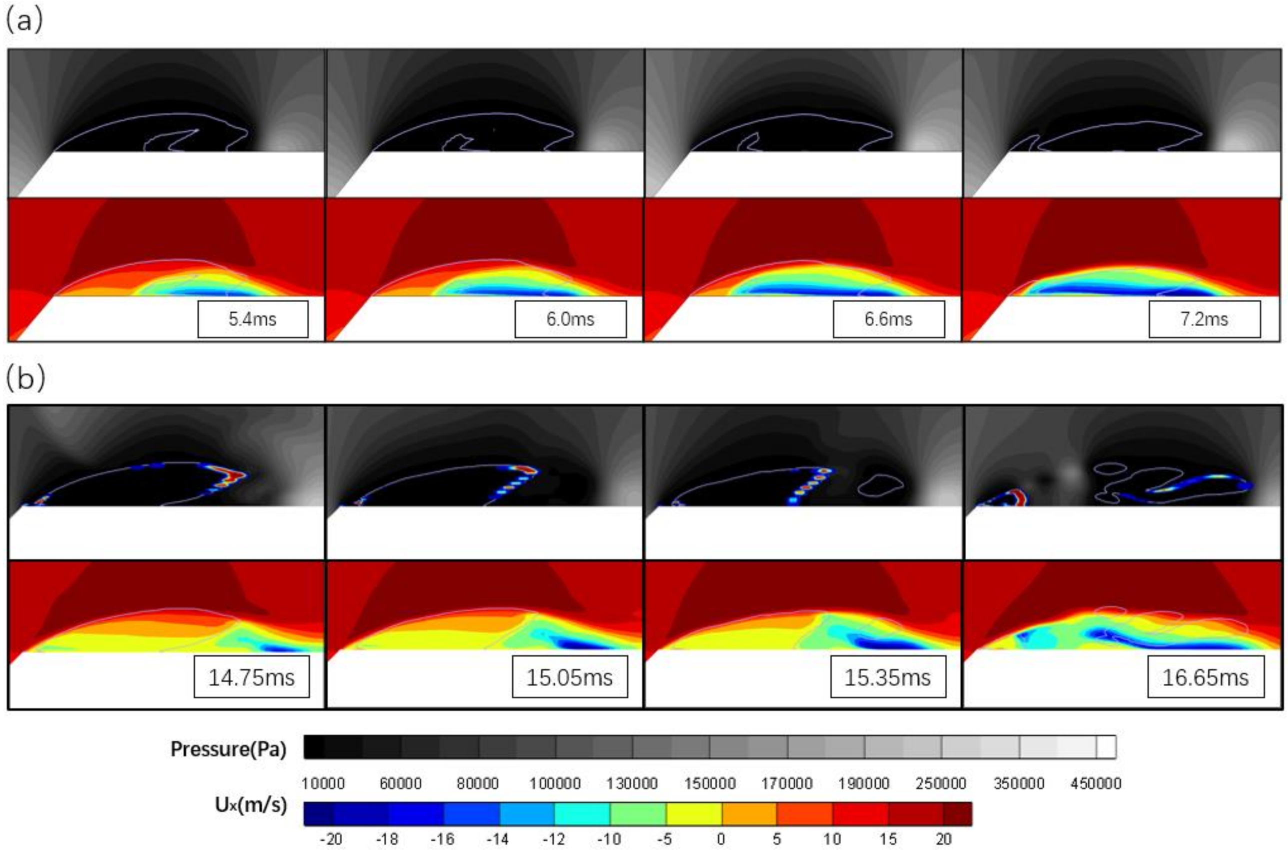

Figure 3: Pressure fields with condensation rate and velocity fields: (a)the first half cycle, (b)the last half cycle

Acknowledgements

The authors thank all those involved in the organisation of OFW13 and to all the contributors that will enrich this event. This research was sponsored by the National Natural Science Foundation of China through grant numbers 11772340 & 11672315. This project was also supported by the Youth Innovation Promotion Association of CAS (2015015)

References

[1] H. Ganesh. Bubbly shock propagation as a cause of sheet to cloud transition of partial cavitationand stationary cavitation bubbles forming on a delta wing vorte. PhD thesis, University of Michigan, 2015.

[2] A. Gnanaskandan, K. Mahesh, A numerical method to simulate turbulent cavitating flows, International Journal of Multiphase Flow, Volume 70, 2015.

[3] F. Moukalled, M. Darwish, A High-Resolution Pressure-Based Algorithm for Fluid Flow at All Speeds, Journal of Computational Physics, Volume 168, Issue 1, 2001.

[4] C. E. Brennen. Cavitation and Bubble Dynamics, USA:Oxford University Press, 1995.

[5] Arndt, Song et al. Instability of partial cavitation- A numerical & experimental approach. Symposium on Naval Hydrodynamics, 599-615, 2001.